International Journal of Mechanical Engineering and Technology (IJMET)
Volume 10, Issue 1, January 2019, pp. 434–449, Article ID: IJMET_10_01_045
Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=10&IType=1
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

EXPERIMENTAL AND SIMULATION INVESTIGATION FOR PERFORMANCE OF A SMALL-SCALE MODEL OF BARE AND SHROUDED HAWT

Balasem A. Al-Quraishi
Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia (UTHM), BatuPahat, Johor, Malaysia
Eng. Technical College -Najaf, AL-Furat AL-Awsat Technical University, Najaf, Iraq

Nor Zelawati Binti Asmuin
Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia (UTHM), BatuPahat, Johor, Malaysia

Mohammed Najeh Nemah
Eng. Technical College -Najaf, AL-Furat AL-Awsat Technical University, Najaf, Iraq

Salih Meri A
Faculty of Mechanical and Manufacturing Engineering,
University Tun Hussein Onn Malaysia (UTHM), Batu Pahat, Johor, Malaysia

ABSTRACT

The HAWTs are existed in different sizes and they used in a different performance requirements depending on the applications variety. In this paper, a small scale of HAWT in scale of 1:6.5 of original model was presented. The scale model of HAWT and a suitable diffuser as a shroud for its, were fabricated and experimented in the wind tunnel in two forms, bare HAWT and shrouded HAWT. the experimental tests were conducted using a new method to power calculation at a different wind velocities ranged 5 - 9 m/s with focusing on a two wind speeds, 7 m/s and 8 m/s for performance tests in terms of power coefficient as a function of the tip speed ratio .the results show an increasing in the power coefficient of the shrouded HAWT model around 1.4 to 1.52 compare to the bare model one. On another hand, the experiments results were validated with CFD simulation results for all cases within accepted error ratio. Moreover, CFD provided a clear visualization for the flow across wind turbine.

Key words: Wind Energy harvesting, HAWT, Shrouded Wind Turbine.

Cite this Article: Balasem A. Al-Quraishi, Nor Zelawati Binti Asmuin, Mohammed Najeh Nemah and Salih Meri A, Experimental and Simulation Investigation For Performance of A Small-Scale Model of Bare and Shrouded HAWT, International Journal of Mechanical Engineering and Technology, 10(1), 2019, pp. 434–449.
http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=10&IType=1

1. INTRODUCTION

Wind power systems, represented by a wind turbines have been the focus of interest of scientists and researchers in the past decades. Flowing of wind through the turbine rotor leads to the production of mechanical energy that can be used in many applications especially to produce electricity. However, power produced by wind turbine is dependent on the Betz limit; an ideal type can extract only 59.3% of incoming energy in stream-tube by turbine blades [1], [2]. Various types of wind turbine exist in different sizes, Horizontal axis wind turbine (HAWT) is one of these types. HAWT that have a rotor diameter of 1.25 m or less called a micro wind turbine. This type of turbine can be used to produce power in urban areas [3], [4]. Several studies have been conducted to predict and develop the performance of HAWT. Some of these studies have been carried out by practical tests; others have been analyzed using programs such as CFD where some of these studies were validated with experiments results.

Despite the development of the design of the turbines, there is an issue that needs to be considered in depth - the economic feasibility of the performance tests. If a small prototype of the turbine can be used for the tests of performance has the same shape and specifications of the large one, the cost can be reduced so that the design can be easily improved. Use a scale down model of wind turbine in a wind tunnel to investigate the performance and other aerodynamic characteristics providing the time and less cost and risks [5]. The effects of wind turbine size on aerodynamic characteristics of a rotor blade are examined by Mohammad Giahi et al [6] using CFD simulation. To ensure the accuracy of the CFD model and validate the results, NREL VI was simulated wind turbine was simulated. A scaled-down model for a large turbine (2 MW) based on its rotor diameter was simulated numerically. CFD simulation showed identical results to the expected values in the traditional similarity. Two models were presented for MARIN Stock Wind Turbine in 1/130th-scale and 1/50th-scale for (NREL) 5 MW. The results provided demonstrate that developmental testing of a performance-matched wind turbine can be performed efficiently at a small, 1/130th scale, and easier to predict of turbine performance [7].

For testing a wind turbine unit in a wind tunnel smaller than actual turbine, a small scale model must be used suitable for wind tunnel size. It is not practical to depend the scaled model on matching Reynolds number. Where if an actual turbine is working at a certain speed as (V), the wind speed on the turbine model used should be greater than (V) in a value of scaling factor (V* SF), so it to looking for other criteria for testing [8].

The conventional method to scaling of wind turbine is based on dimensional matching without considering of Reynold number, where a scale factor is considered for all dependent variables [9].Despite the simplicity of the theory of similarity, it is very effective to investigate the scaling effect on the properties of the turbine. based on this theory, the original and scale models are consider have same flow conditions if a three conditions are met , (i) both models have same TSR, (ii) same profile of blade and blades number for both models, and (iii) make an adjustment in proportional to dimensions, like rotor radius [5], [10].

Experimental tests on a fabricated scaled (1:13) model of HAWT in wind tunnel has been conducted at CSIR-SERC, Chennai by K. Sankaranarayanan et al [11], where the model has been designed using Computer Aided Design and Drafting (CADD) based on BEM theory (twisted blade from root to tip). The rotor torque was measured at a different range of wind

speeds 3-9 m/s for calculated tip speed ratio. The optimum tip speed-ratio 6 was achieved at wind speeds of 7- 8 m/s.

Improving of HAWT performance can achieved by shrouded the turbine in a duct (diffuser). Many studies were conducted in this field, the augmentation of power factor was achieved of 2 to 5 [12] and around 1.5 to 2 [13]. The power coefficient and optimum TSR for shroud turbine much higher than bare. The reason of TSR increase because increase of mass flow rate through rotor plane, where this increase produced from the pressure drop at diffuser outlet, so the rotor need to rotating with a higher rpm [12]. 3 kW shrouded HAWT has been developed by K. Olasek et al [14].

The model of the turbine that used in the experiments was in Scaled of 1:6 according to the size of the available wind tunnel. The experimental results showed that C_P of shroud turbine up to 70- 75 % more compared with a bare rotor. A small-scale wind energy portable turbine (SWEPT) targeted to operate below 5 m/s wind speed has been tested in a wind tunnel. The maximum coefficient of performance of 14% was recorded at the tip speed ratio of 2.9. It was observed through CFD simulation that more power can be produced up to 1.6 if SWEPT is shrouded in a diffuser [15].The results of experiments and CFD simulation for HAWT (190 mm of rotor diameter), has been showed an increasing in C_P up to two times if its shrouded in a diffuser. On another hand, the shroud is valid even for a high freestream turbulence[16].

A small wind turbine with a simple frustum diffuser has been simulated by CFD at different sizes of diffuser length (L/D) in range of (0.1-0.4) and area ratio. The results showed that increase of area ratio with constant L lead to the expansion of flow through diffuser causes decreasing of downstream pressure and the suction side of rotor blades, which create lift force on the blades, hence increase the rotor power [17].

There are a different methods were used to conduct the experimental tests for the wind turbines performance. All of these methods focused to calculate the power through knowing the rotor torque and angular velocity. Many researchers use an AC or DC motor to control the rotational speed and used a torque sensor to measure the torque at each rpm recorded e.g [16], [18]. An electrodynamometer was connected to turbine shaft via a belt drive to Measure of the shaft speed and torque in order to calculate the power transmitted [8]. The torque of wind turbine was measured using de Prony brake assembly depending on the product frictional force and radius of the rotor shaft [11].

In the present paper, an experimental test was carried out to study the performance of a small scale model of bare HAWT model. The experiments performance was in terms of torque, power, and power coefficient, which was validated with CFD results. The study also included a comparison of a small scale of bare HAWT performance with the same HAWT shroud within a suitable diffuser. A new method was used to test HAWT performance experimentally based on calculation the power using a load control board. This paper aims to investigate effect of the scaling on the HAWT performance tests, then investigation in performance of small scale model of same HAWT, if it is shrouded by a suitable diffuser without flange.

2. METHODOLOGY

2.1. Simulation Description

In this study, 3D CFD model was used (virtual wind tunnel) based on dimensions of the test section of Wind Tunnel (125 x 40 x 40 cm) in aerodynamic Lab, Faculty of Mechanical Engineering and Manufacturing (FKMP), University Tun Hussein Onn Malaysia (UTHM) as shown in Figure 1 (a) [19] to investigate the turbine rotor with and without diffuser. The

model of wind turbine rotor used in this study is a small-scale (1:6.5) of full the turbine rotor presented by M.-H. Lee et al[20], with three blades of NACA SD8000. Thus, the parameters design of turbine model is shown in Table 1 and the section design for the blade is listed in Table 2.The diameter and length of diffuser that used to shroud the turbine are 16 cm and 8 cm respectively with expansion angle of 12◦ without flange and 3mm wall thickness presented by [19].The diffuser and rotor models were modeling by solid work in same model dimensions.

Table 1 The main parameters of rotor design

Rotor diameter (D_r)	15.4 cm
Hub diameter (H_r)	2.46 cm
Span radius (l)	6.16 cm
Number of blades (B)	3
Number of elements per blade	10
element radial distance (dr)	0.77 cm
Airfoil type	NACA SD8000
Angle of attack (α)	5°
Design tip speed ratio (λ)	5
Design wind velocity (V_∞)	10 m/sec

Table 2 Design parameters of the geometry of the blade.

section	R/r	Chord length (cm)	Pitch angle (deg)
1	0.2	2.36	25
2	0.29	2.05	18.1
3	0.38	1.74	13.6
4	0.47	1.48	10.5
5	0.56	1.29	8.2
6	0.64	1.14	6.5
7	0.73	1.02	5.2
8	0.82	0.91	4.1
9	0.91	0.83	3.3
10	1	0.77	2.5

Since the wind power is extracted by the turbine rotor which is convert this power to a mechanical power. Thus the mechanical power (P_{out}) of wind turbine is the product of rotor torque (Q) and angular velocity (Ω) of rotor and it can be easily calculated if we can accurately determine the torque and angular velocity of the rotor when it is running without any external load.

The wind turbine performance is represented by the power coefficient (C_P) which is defined as:

$$C_P = \frac{Q\Omega}{0.5\,\rho\,A\,V_\infty^3} \qquad (1)$$

The most common method used to preview the power coefficient by graphing its values as function with Tip speed ratio (λ) which is defied as:

$$\lambda = \frac{Q\mathrm{R}}{V_\infty} \qquad (2)$$

Where Q is rotor torque, Ω is angular velocity, ρ air density, A is rotor swept area, V_∞ is upstream wind velocity, and R is rotor radius.

For high accuracy of flow investigation, the numerical model in a 3- D domain by using CFX was carried out for a full geometries of the turbine rotor with and without diffuser model and the virtual test section of the wind tunnel (FKMP, UTHM). The mesh type was Tetrahedron with (2162219) elements and (471644) nodes for rotor model while numbers of elements and nodes for model of rotor with diffuser were (4249560) and (843992) respectively. The skewness was (0.7- 0.8) and orthogonal quality (0.999) (see Figure 1).

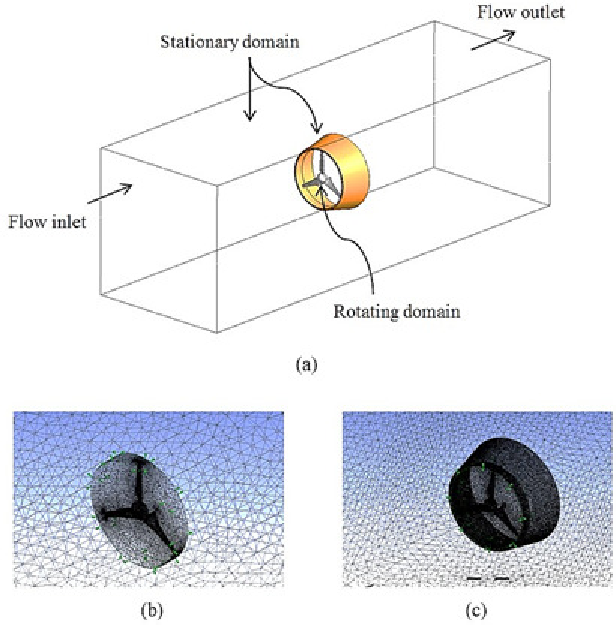


Figure 1. (a) CFD domain. (b) Mesh domain of bare turbine. (c) Mesh domain of shrouded turbine

The simulation was carried out in a steady -state using ANSYS 19.1. The flow was treated as isothermal .The working fluid is air as ideal gas, and the reference pressure is1atm at default turbulence intensity [19]. The computational field consists of two regions, a stationary and a rotating domain. The stationary domain is wind tunnel included the diffuser. The rotating domain included turbine rotor enclosed in the rotating frame (disc). The support system of the rotor inside the diffuser was not included. The interface model between the stationary and rotating frames that adopted is frozen rotor with automatic pitch change[16], [21]. This model is dependent a special scheme for moving and nonmoving zones for both frames also it is provide a transferred for absolute velocities based on global coordinate system. The wall of tunnel, surfaces of the diffuser's walls, the rotor blades and the hub of the rotor are considered as no slip walls. At the inlet of the computational field the wind velocity was fixed and at the outlet a relative static pressure equal to 0 Pa was set. The rotating domain set to the selected rotational velocity. The turbulence model which adopted is (SST *k-w*), which give a very good prediction for the results [22] as well as it has been used by [16], [21], [23] and [24]. The residual target of convergence criterion is set at 10^{-4}.

2.2. Model verification

The verification of CFD simulation results which has been carried out by M.-H. Lee et al [20] was done, first, by using same previous model in scale 1:1 at same conditions .The simulation to verify the model was conduct at wind speed of 8 m/s at different range of revolution per minute. The verifying results of the power coefficient as a function of tip speed ratio is shown in Figure 2 with accepted error ratio.

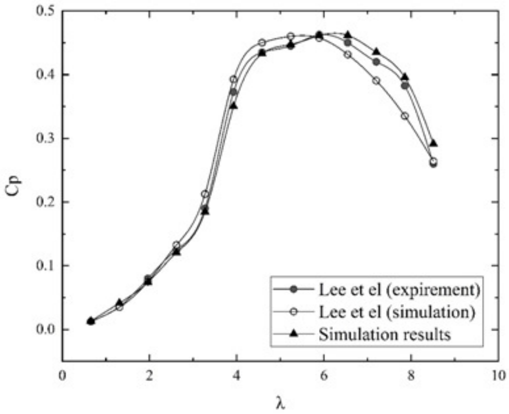


Figure 2. Verification curves for 3-D simulation results of previous turbine design by (M.-H. Lee) in scale (1:1) at wind velocity of 8m/sec.

As mentioned above, many researchers have focused on selecting a scaled model for performance tests of a wind turbine. Some of them adopted on the matching of Reynolds Number, while the other adopted on the similarity of the form which depended in this study according to the available size of wind tunnel whether real or virtual [8].

Although there is no fixed base in this type of match, there are studies that gave predictions closer to the real values of the performance tests, where it was assumed that there is a scale factor (SF) considered as a function for most of operation components as, Maximum power, torque and rpm. The SF defined as the ratio of the original turbine diameter to the diameter of the scale model (D_o/D_m). So, in this study SF was adopted a function for maximum angular velocity, torque and power of wind turbine, which is within the used ranges in previous studies as shown in Table 3. In another hand, there is a large loss in performance parameters (power coefficient, C_P, and thrust coefficient, C_T) for the scale model turbine as compared to the desired full-scale performance [7]. As example, testing of NREL 5MW model in SF of (1:50) at MARIN in 2011, which had achieve the peak C_P of 0.04 at tip speed ratio of 3.91, while the original model achieved maximum C_P of o.47 at tip speed ratio of 7.5 (see Figure 3).This decline in expected performance due to Reynolds number Mismatch between the original and scale model [25].

Table 3 Performance-matched turbine design criteria.

Author	Scaling	Blade number B_o/B_m	Blade chord C_o/C_m	rpm Ω_o/Ω_m	Torque Q_o/Q_m	Thrust T_o/T_m	Power P_o/P_m
[6]	SF	1	1	1/SF	$(SF)^3$	$(SF)^2$	$(SF)^2$
[10]	SF	1	1	1/SF	$(SF)^3$	$(SF)^2$	-
[7]	λ	1	-	1/ λ	-	-	-
present study	SF	1	1	1/SF	$(SF)^{3.2}$	-	$(SF)^{2.2}$

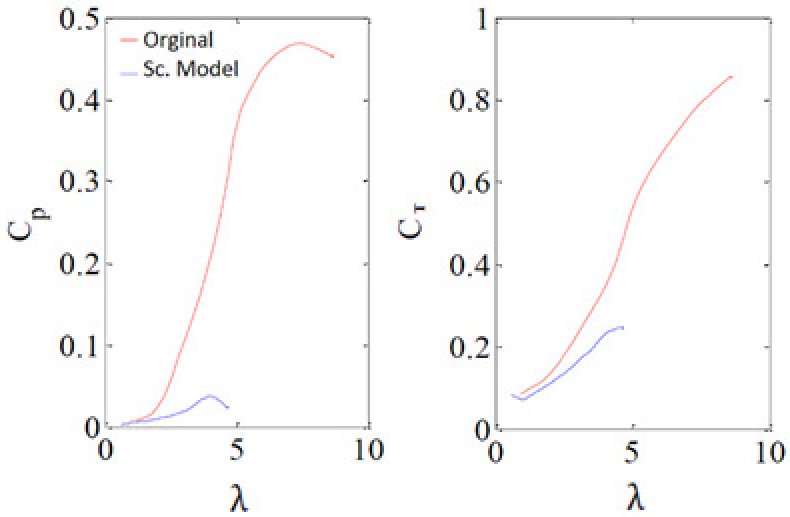


Figure 3. C_P and C_T of NREL 5MW model in scale of 1/50 model as tested at MARIN [7].

On this basis, the new 3D CFD model that shown above in Figure 1 was also used to verifying with previous study model. The results of verifying shown in Figure 4, explain that C_P for the present (1:6.5) scaled model of wind turbine used in this study is less than C_P for original (large size) wind turbine but they have appear at same behavior.

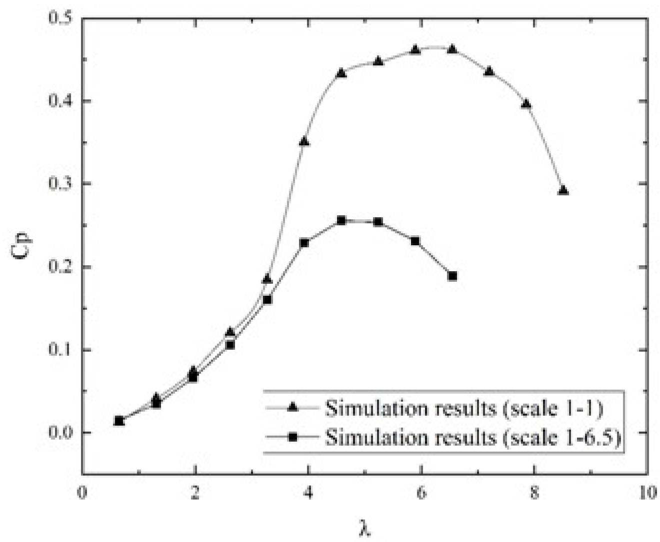


Figure 4. CFD comparison of power coefficient for the small turbine model to original model

2.2. Experimental set up

For study the performance of the turbine accurately, it is necessary to conduct experiment tests of the models of turbine rotor with and without diffuser. The prototypes of rotor (containing three blades and hub) and diffuser were modeled using solid work and manufactured from RGD450 material using polyjet 3D printing technology as describe in Figure 5. The wind tunnel of FKMP, UTHM with test section of 125 cm × 40 cm × 40 cm is used which can produced a maximum wind velocity of 40 m/s. The turbine is placed at a distance of 60 cm from inlet. The turbine is attached to a 3-phase AC 12 Watt alternator generator which connected to a 6- diode full wave rectifier to convert the suppling power to a 1-phase DC [20]. Since the 3-phase AC supply provides high power as the required and it has a fixed voltage and frequency it can be used by a rectification circuit to produce a fixed

voltage DC power. Therefore, the wind energy will convert to a directly proportional measurable electrical power. Thus, it can be recorded and analyzed easily.

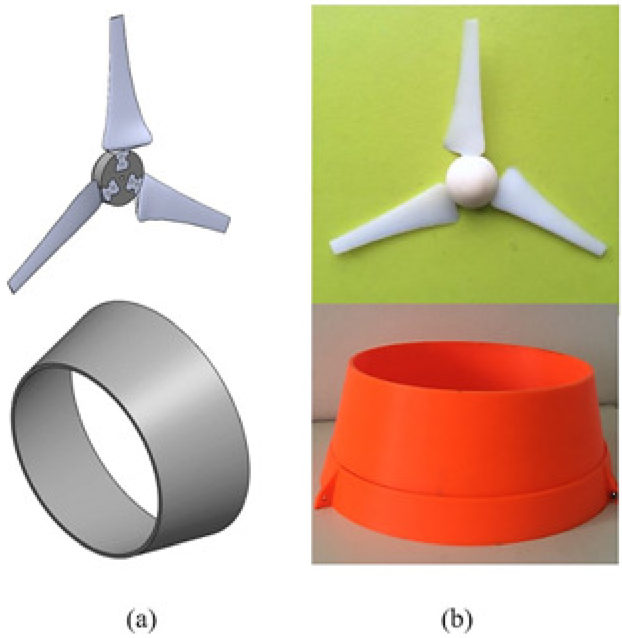


Figure 5 Rotor and diffuser models used (a) modeling models (b) prototype models

The FLUKE 922 digital Airflow Meter with Pitot tube used to measure the velocity and pressure of inlet air [26]. The angular velocity of the wind turbine was measured using non-contact type, digital Tachometer ‘DT-2234B’ (RS 445-9557) [27].

A load control board for the wind turbine power calculation was designed and fabricated, as shown in Figure 6, in order to face the challenge of difficulty measure the HAWT power. In this case, the measurable electrical power will estimate the mechanical power of the turbine. The main idea of this board is to apply constant loads on the turbine, step by step, and calculate the turbine power at each round. Accordingly, 16 of 12V, 1 Watt LEDs load were utilized and connected as a series electrical circuit. While a 16 channel relay module board with an external 12V power supply were used to automatically perform the loading and non-loading states [28]. In addition, Arduino MEGA 2560 microcontroller used to control the operation of the relay board, for enabling the experimenter from adjusting the experiment's time constant. The time constant is a delay time in an electrical loading circle. In other words, it is the time that is used from the experimenter to record the measuring data before increasing or decreasing the load. In general, if L is the resistance of each single LED load and n is the total number of the using LED loads, which is equal to 16. Thus, the total load that applying on the wind turbine generator (L_{total}) can be calculating as follow:

$$L_{total} = \sum_{n=1}^{16} n * L \tag{3}$$

Finally, the Matlab 2018 program modified with Arduino Simulink toolbox was utilized to interface the digital circuit with the computer and control it. Moreover, tow of ANEGN AN8002 AC/DC Digital Multi-meters connected directly across the output power of the generator (the input power to the load control board) to record the voltage and the current for each loading case [29],where the power generated is equal to multiplier voltage into the current. At the experiment, the experimenter has to set a suitable time constant, at which he will be able to record the rotational speed, voltage, and the current before the next load will be auto applying.

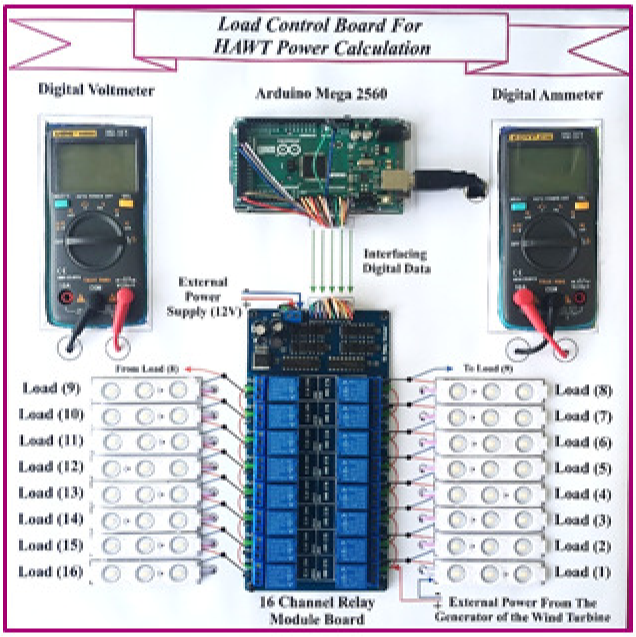


Figure 6 The load control board for HAWT power calculation

The experiment wind tunnel schematic diagram and the experimental setup are shown in Figure 7 and Figure 8 respectively. The generator of the turbine is connected to load; the voltmeters were used to measure the voltage and current across the load. By maintaining air velocity is constant, the load can be varied by operating more load (lamps), the voltage and current were recorded at each load. At the same time, the angular velocity is recorded at each load. The experiment was repeated several times based on cases of the models, turbine rotor with and without diffuser. The experiments for the two models were conducted at a two wind speeds 7 and 8 m/sec.

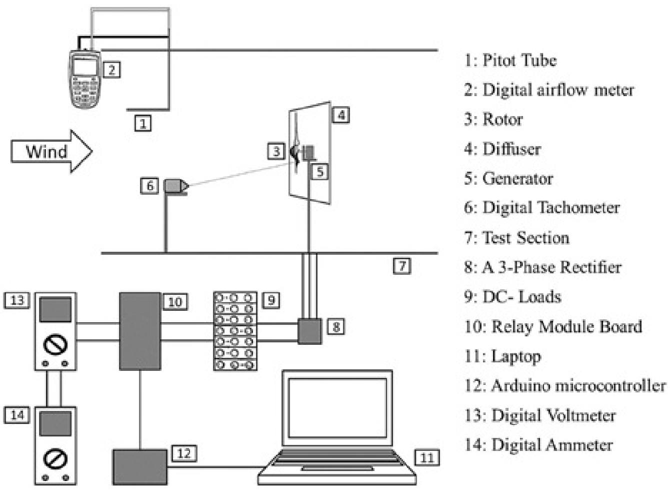


Figure 7 The schematic diagram of experimental set up.

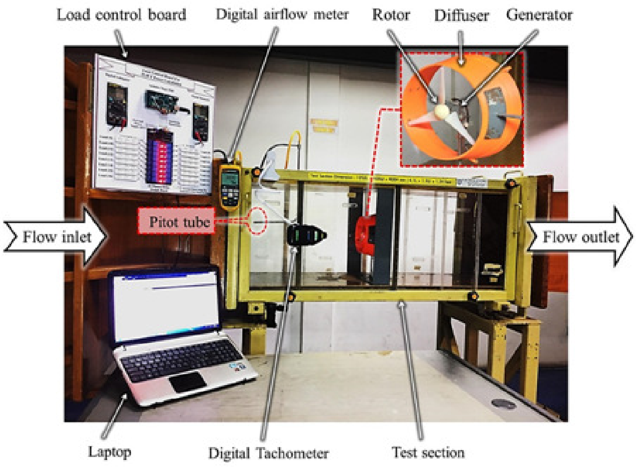


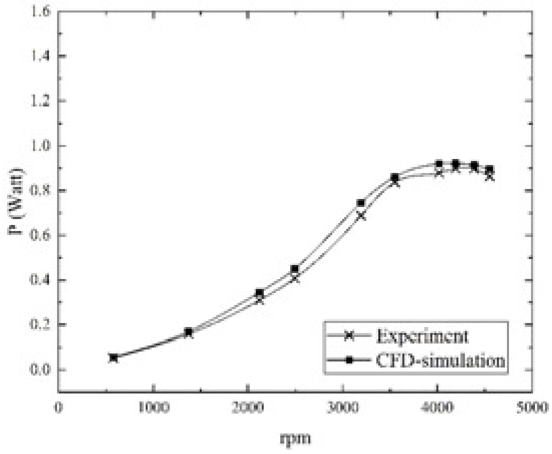
Figure 8. The experimental set up of testing the HAWT.

3. RESULTS AND DISCUSSIONS

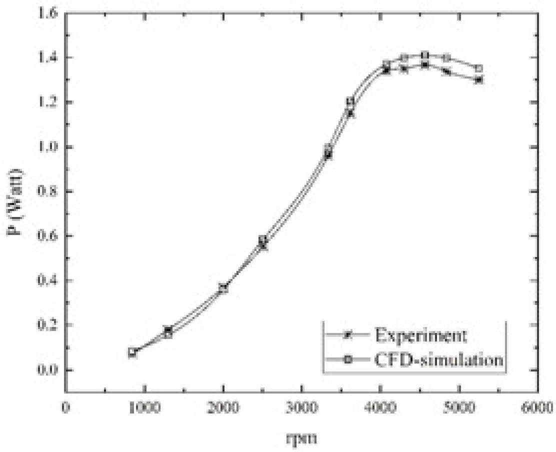
The 3D CFD for an empty diffuser (without flange) have been conducted using stationary model that presented in previous study for an empty diffuser [19] , the results show an increasing in flow velocity at 1 cm (0.0625 D, a distance from diffuser inlet) along the position of rotor radius (7.7 cm). The simulation was conduct at a different range 1-15 m/s of inlet flow velocities. The average velocity increase was 1.256.

3.1. Mechanical Power of Wind Turbine

As mention previously, to calculate the power extracted by the wind turbine, must be know a two components are torque and angular velocity of the rotor. The angular velocity can be measured experimentally using tachometer in rpm, but not easy to measure the rotor torque due to the high cost of the torque sensor device especially for a small turbines [15], thus an alternative experimentally method was used in this study. (P_{out}) of bare and shrouded HAWT was calculated from the voltage and current recorded at each load with considering values for the mechanical and conversion efficiencies for the generator. For calculation the rotor torque from CFD simulation results at each angular velocity recorded, an expression is used which available in CFD-post.The experimental tests and CFD simulation for the bare and shrouded wind turbine were conducted at two velocities 7 and 8 m/s. the results showed a significant increase of power of shrouded turbine as compare with the bare one. Figure 9 (a) and (b) show the power curves for bare and shrouded turbine at upstream wind velocity of 7 m/s. it can see the power is increased with rpm increased until reached to maximum power value. The maximum power value recorded experimentally for bare turbine was about 0.9 Watt while, it reached to 1.36 Watt for shrouded turbine. In other words, the power increased by 52%.



(a)



(b)

Figure 9. The output power of as function of rpm at wind velocity of 7 m/s (a) Bare HAWT, (b) Shrouded HAWT.

3.2. The performance of Wind Turbine

Since power coefficient indicate to the wind turbine performance, so the results of experiments and CFD for Bare and Shrouded HAWT focused to measure C_P at varies range of tip speed ratio at different rotational speed. The measured C_P values of the both models are plotted as a function ofλ in Figures 10 and 11 forV_∞= 7 m/sand 8m/s, respectively. As can be seen from these plots, the shrouded turbine rotor has much higher C_P than those of the bare turbine rotor at these wind speeds .This is because presence of diffuser which increase the wind velocity at rotor plane, these increase led the rotor rotate faster, hence the rotor torque increase to some extent. The trends of the both Figures 8 and 9 are a same, but still there is a little difference. At V_∞ = 7 m/s, the shrouded rotor has the maximum $C_P$of 0.372 at λ = 5.253which is pretty close to the designed value ofλ. While the bare rotor has maximum value of C_P = 0.25114 at λ = 4.827. At V_∞ = 8 m/s the maximum C_P for shrouded and bare rotor are 0.380 and 0.253 at λ of 5.232 and 4.717 respectively. In addition, it can be noted from the graphs there are a well pretty agree for the CFD simulations with the experiments data.

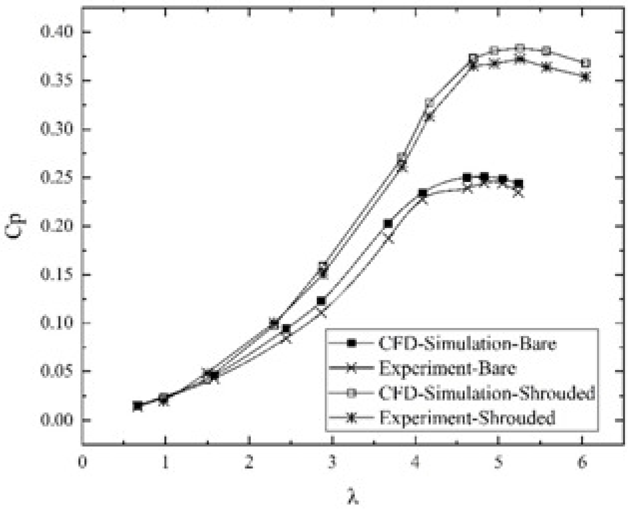


Figure 10. The power coifficient of Bare and Shruded HAWTat V= 7m/s.

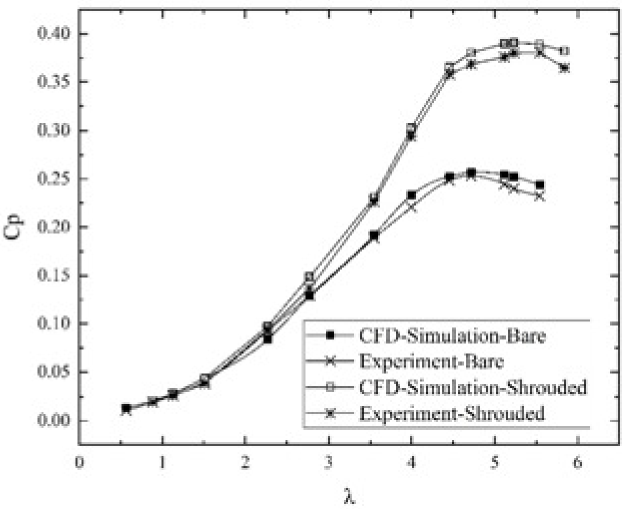


Figure 11. The power coifficient of Bare and Shruded HAWTat V= 8m/s.

The maximum mechanical power for the Bare and shrouded turbine as a function of wind velocity for a range from 5 - 9 m/s is shown in Figure 12. This range of velocities have been considered due to the generator limits, unable to convert all of the generated torque on the rotor to mechanical power. On other hand, the velocities that below of 5 m/s unable to rotate rotor in case of connect a load to the generator. It is interesting to note that even though the maximum power out increases with increase of wind velocity but it is always look same increase for all wind velocities for the two models, the bare and shrouded HAWT. In another hand, Figure 12 also show the power augmentation (P_{shroud} / P_{bare}) in the range of 1.46 - 1.52.

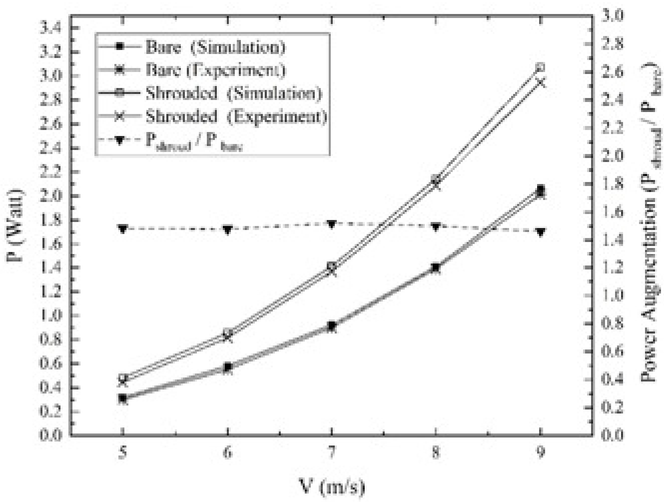


Figure 12 Mechanical Power of Bare and Shrouded HAWT at different wind velocities

In addition, the CFD simulation provide a good vision for investigate the velocity and pressure for the empty diffuser and HAWT at $\mathbf{V}_{\infty}$= 7m/s. Figure 13 show the velocity and pressure contour for the empty diffuser, Figure 14 show the velocity flow at HAWT with and without diffuser. Figure 15 and 16 show a compression of the pressure contours for bare and shrouded HAWT. Figure 17 show the stream line of flow for bare and shrouded HAWT. It can be observed that the pressure decrease inside the diffuser cause an increase in flow velocity hence, more mass flow through the rotor plane which make the rotor is rotating faster.

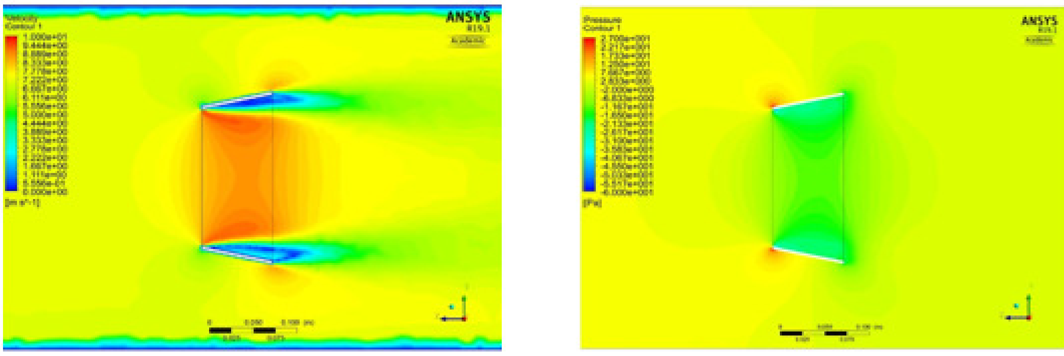
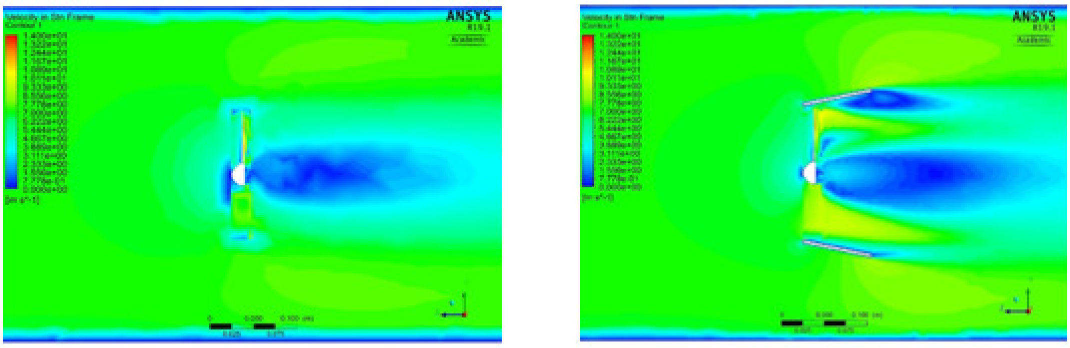
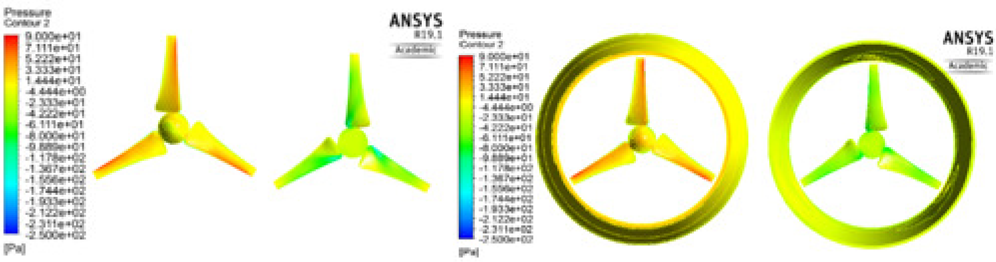


Figure 13. Empty diffuser without flange at V_∞=7m/s(a)velocity contour, (b) pressure contour.



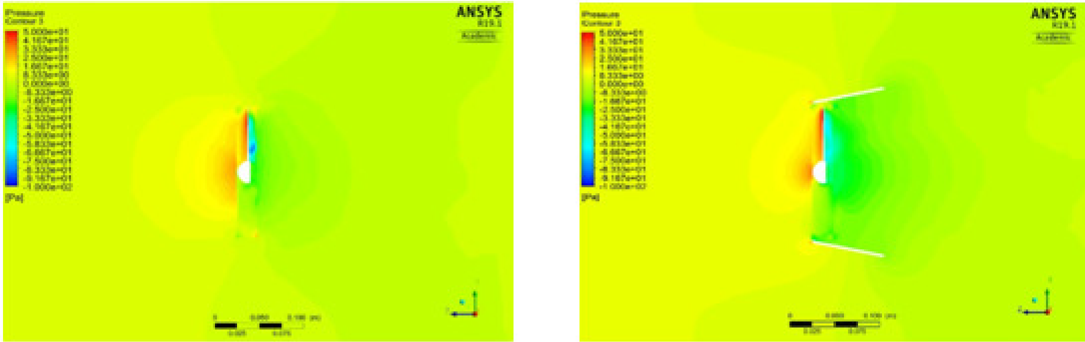
(a) (b)

Figure 14. Velocity contoure at V_∞=7m/s, (a) bare HAWT, (b) shrouded HAWT



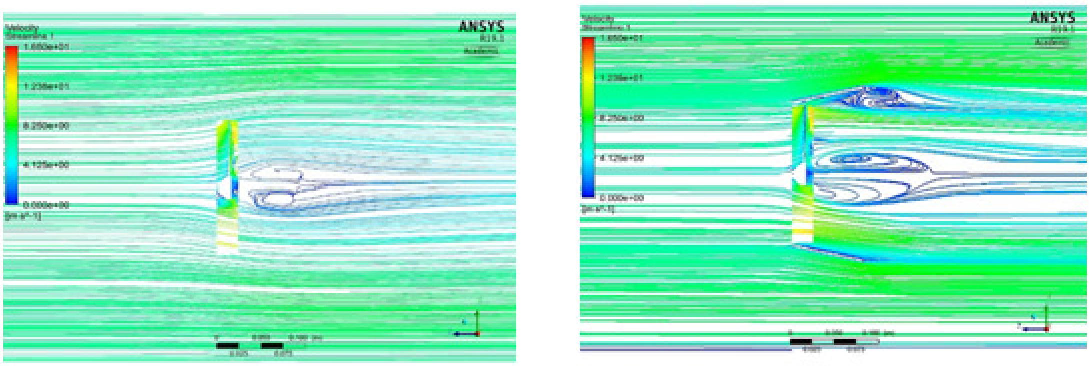
(a) (b)

Figure 15. Rotor pressure distribution at V_∞=7m/s(a) bare HAWT, (b) shrouded HAWT



(a) (b)

Figure 16. Pressure contoure at V_∞=7m/s(a) bare HAWT, (b) shrouded HAWT



(a) (b)

Figure 17. Stream line contour at V_∞=7m/s(a) bare HAWT, (b) shrouded HAWT

4. CONCLUSION

In this paper, a small scale model of HAWT has been presented in scale of 1:6.5, which was verify with the original model in term of C_P using CFD. The scaled model of HAWT was modeled with considered maintain TSR, profile of blade and blades number. The model was fabricated and tested with and without diffuser in the wind tunnel. the experimental performance tests were conducted using a new method based on a load board control to calculate voltage and current produced from the generator at each round (load add). The performance of HAWT with and without shroud was investigate at a two wind velocities 7 m/s and 8 m/s in terms of power as a function to rpm and C_P as a function to TSR. The results show that a significant increase in the C_P for shrouded HAWT compare with the bare model one with maintain the range of tip speed ratio pretty close to the designed value. moreover, the maximum mechanical power of bare and shrouded HAWT was calculated for a different wind velocities ranged of 5 - 9 m/s, the results show an increasing in power with increased of wind velocity but this increase look in same behavior at all velocities range for the two models (the bare and shrouded HAWT) .On another hand, the results show that the augmentation of power (Pshroud / P bare) was in the range of 1.46 - 1.52. The experiments results for all cases were validated with CFD simulation within accepted error ratio. Also the CFD provided a visualization for the flow across wind turbine.

REFERENCES

[1] J. N. Libii and D. M. Drahozal, "The influence of the lengths of turbine blades on the power produced by miniature wind turbines that operate in non-uniform flow fields," World Trans. Eng. Technol. Educ., 10, no. 2, 2012, pp. 128–133.

[2] O. IGRA, "Research and Development for Shrouded Wind Turbines," *Energy Convers. Manag.*, 21,1981, pp. 13–48,.

[3] A.D. Peacock et al "Micro wind turbines in the UK domestic sector", Wind Energy Build, 2008, pp.1324-1333.

[4] P. Gipe, Wind Energy Basics, Chelsea Green Publishing Company, Vermont, Totnes, England, 1999.

[5] R. Gasch, J. Twele, "Scaling wind turbines and rules of similarity", Wind Power Plants, Springer Berlin Heidelberg, pp. 257-271, 2012.

[6] Mohammad H. Giahi, Ali J.D, Investigating the influence of dimensional scaling on aerodynamic characteristics of wind turbine using CFD simulation, Renewable Energy, 97, 2016,pp. 162-168.

[7] Richard K. et al, Wind/Wave basin verification of A performance-matched scale-model Wind Turbine on a Floating offshore Wind Turbine Platform, Proceedings of the ASME 33rd International Conference on Ocean, Offshore and Arctic Engineering, San Francisco, California, USA, 2014.

[8] R. F. Ghajar and E. A. Badr, "An Experimental Study of a Collector and Diffuser System on a Small Demonstration Wind Turbine," *Int. J. Mech. Eng. Educ.*, 36, no. 1, 2008,pp. 58–68.

[9] D.T. Griffith, T.D. Ashwill, "The Sandia 100-meter All-glass Baseline Wind Turbine Blade": SNL100-00, Technical report, Sandia National Laboratories, California, United States, 2011.

[10] Kenneth W. V. T., Timothy A. B., Experimental testing of wind turbines using wind tunnels with an emphasis on small-scale wind turbines under low Reynolds numbers, Ch-4, Department of Mechanical Engineering, Baylor University, Waco, TX, USA, WIT Press, 81, 2014.

[11] K. Sankaranarayanan, S. Krishnakumar, G. V. PaulRaj, and R. Rahul, "Wind Tunnel Experiment on a Small Horizontal Axis Wind Turbine," *Eighth Asia-Pacific Conf. Wind Eng. ,Chennai, India*, 2013.

[12] Y. Ohya, T. Karasudani, "A shrouded wind turbine generating high output power with wind-lens technology", Energies. 3, 2010, pp. 634-639.

[13] B. Kosasih, A. Tondelli, "Experimental study of shrouded micro-wind turbine", Procedia Eng., 49, 2012, pp. 92-98.

[14] K. Olasek, M. Karczewski, M. Lipian, P. Wiklak, and K. Józwik, "Wind tunnel experimental investigations of a diffuser augmented wind turbine model," *Int. J. Numer. Methods Heat Fluid Flow*,26, no. 7, 2016, pp. 2033–2047.

[15] Ravi A.Kishore, Thibaud Coudron, Shashank Priya, Small-scale wind energy portable turbine (SWEPT), J. Wind Eng. Ind. Aerodyn. 116, 2013, pp.21–31.

[16] B. Kosasih, H. Saleh Hudin, Influence of inflow turbulence intensity on the performance of bare and diffuser-augmented micro wind turbine model, Renewable Energy 87 ,2016, pp.154-167.

[17] S. A. H. Jafari, B. Kosasih, Flow analysis of shrouded small wind turbine with a simple frustum diffuser with computational fluid dynamics simulations, J. WindEng.Ind.Aerodyn.125, 2014, pp.102–110.

[18] K. Abe et al, Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser, Journal of Wind Engineering and Industrial Aerodynamics,93, 2005, pp. 951–970.

[19] Balasem A. Al-Qureshi et al, CFD Investigation of Empty Flanged Diffuser Augmented Wind Turbine, Proceedings toInternational Conference on Energy Systems andEnvironmental Management(ESEM), Thailand, 2018.

[20] M. H. Lee et al, Experiments and numerical simulations of the rotor-blade performance for a small-scale horizontal axis wind turbine, J. Wind Eng.Ind.Aerodyn.,149, 2016, pp.17–29.

[21] A. Tourlidakis et al, Aerodynamic design and analysis of a flanged diffuser augmented wind turbine, Proceedings of ASME Turbo Expo: Turbine Technical Conference and Exposition, San Antonio, Texas, USA, 2013.

[22] F. R. Menter, “Two-equation eddy-viscosity turbulence models for engineering applications,” *AIAA J.*, 32, no. 8, 1994, pp. 1598–1605.

[23] H. M. Elbakry et al,“Simulation ofDiffuser Augmented Wind Turbine Performance,” *World Congr. Sustain. Technol.* 2016, pp. 40-48.

[24] L. Michal, K. Maciej, and J. Krzysztof, “Analysis and comparison of numerical methods for design and development of small Diffuser-Augmented Wind Turbine (DAWT),” *IECON Proc. (Industrial Electron. Conf.*, 2016, pp. 5525–5531.

[25] Heather R. Martin et al, Methodology for Wind/Wave Basin Testing of Floating Offshore Wind Turbines, Journal of Offshore Mechanics and Arctic Engineering, 136, 2014.

[26] Fluke Corporation,“Airflow Meter 922”, PN 2683880, users manual, November 2006.

[27] RS PRO, “Laser, Photo Tachometer”, RS 445-9557, datasheet.

[28] UCTRONICS, “16- Channel Relay Module Board” datasheet, Rev 1.0, July 2018.

[29] ANEGN, “AN8002 AC/DC Digital Multi-meters” datasheet.